Elasto-plastic Analysis of Plate Using ABAQUS

The thesis submitted in partial fulfillment of requirements for the degree of

Master of Technology

in

Civil Engineering

(Specialization: Structural Engineering)

by

ROHAN GOURAV RAY

213CE2073

Department of Civil Engineering

National Institute of Technology, Rourkela

Odisha, 769008, India

May 2015

Elasto-plastic Analysis of Plate Using ABAQUS

A Dissertation submitted in May 2015

to the department of

Civil Engineering

of

National Institute of Technology Rourkela

in partial fulfillment of the requirements for the degree of

Master of Technology

by

ROHAN GOURAV RAY

(Roll 213CE2073)

under the supervision of

Prof. ASHA PATEL

Department of Civil Engineering

National Institute of Technology, Rourkela

Odisha, 769008, India

DEPARTMENT OF CIVIL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

ODISHA, INDIA

CERTIFICATE

This is to certify that the thesis entitled "**ELASTO-PLASTIC ANALYSIS OF PLATE USING ABAQUS**" submitted by **ROHAN GOURAV RAY** bearing roll number **213CE2073** in partial fulfillment of the requirements of the award ***Master of Technology*** in the Department of Civil Engineering, National Institute of Technology, Rourkela is an authenticate work carried out by him under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other university/institute for the award of any Degree or Diploma.

Place: Rourkela

Date:

Prof. Asha Patel

Civil Engineering Department

National Institute of Technology, Rourkela

DEPARTMENT OF CIVIL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA, 769008

A C K N O W L E D G E M E N T

It gives me immense pleasure to express my deep sense of gratitude to my supervisor **Prof. Asha Patel** for her invaluable guidance, motivation, constant inspiration and above all for her ever co-operating attitude that enable me to bring up this thesis to the present form.

I express my thanks to the Director, **Dr. S.K. Sarangi,** National Institute of Technology, Rourkela for motivating me in this endeavor and providing me the necessary facilities for this study.

I am extremely thankful to **Dr. S.K. Sahu**, Head, Department of Civil Engineering for providing all help and advice during the course of this work.

I am greatly thankful to all my staff members of the department and all my well-wishers, class mates and friends for their much needed inspiration and help.

Last but not the least I would like to thank **my parents and family members**, who taught me the value of hard work and encouraged me in all my endeavors.

Place: Rourkela

Date:

Rohan Gourav Ray

M.Tech., Roll No: 213CE2073

Specialization: Structural Engineering

Department of Civil Engineering

National Institute of Technology, Rourkela

ABSTRACT

Plates and shells are very important parts of engineering structures. The performance of structures depends on accurate assessment of the behavior of all such elements. Accurate valuations of the maximum load the structure can carry, along with the equilibrium path followed in elastic and inelastic range emphasize the importance of material non-linearity to understand the realistic behavior of structures. Modeling elements in the inelastic range incorporating the theory of plasticity is complex and lengthy and demands for heavy computations.

Elastic-Plastic (with or without strain hardening) is a trivial issue in modeling the material (for both uniaxial and multi-axial Von Mises criteria), in numerical procedures like FEM or in modeling using commercial software. ANSYS or ABAQUS contained already defined material subroutines for such behavior.

The objective of this study is to have better understanding how ABAQUS performs nonlinear analyses of plate under uniformly distributed load incorporating material nonlinearity. Two material behaviors are considered perfectly plastic and linear strain hardening. The results are validated with reference data from Owen & Hinton (1980) and results obtained from FEM numerical solution. The results are compared in terms of load deflection diagrams,first yield load, collapse load and plastic or yield flow. The effect of thickness and boundary conditions are also studied.

ABAQUS results and numerical results are found to be in good agreements. The plastic flow patterns clearly depict the perfectly plastic and isotropic strain hardening behaviors and also follow the patterns given by Yield line analysis of slabs. The patters obtained from ABAQUS and numerical solutions are compared and found to be similar.

Contents

CERTIFICATE iii

ACKNOWLEDGEMENT iv

ABSTRACT v

1. INTRODUCTION 2

  1.1 Non-Linearity 3

2. LITERATURE REVIEW 10

3. THEORY AND FORMULATION 13

  3.1 Abaqus Modeling and analysis 13

  3.2 Formulation for Finite Element Method 16

4. RESULT AND DISCUSSION 28

  4.1 Methodology 29

  4.2 Problem Statement 1 31

    4.2.1 Analysis of perfectly plastic material behavior: 31

  4.3 Problem Discussion 2 42

5. CONCLUSION 48

REFERENCES 51

LIST OF TABLES

S. No.	Titles	Page
4.1	Convergence Study	31
4.2	Deflection value at center from ABAQUS, MATLAB and reference data	34
4.3	Yield load and collapse load for simply supported squareplate with varying thickness	37
4.4	Yield load and collapse load for plate having different boundary conditions	40

LIST OF FIGURES

S. No.	Titles	Page
3.1	Linear and nonlinear spring characteristics	3
3.2	Large deflection of cantilever beam	4
3.3	Yield surface for isotropic strain-hardening material	6
3.5	Yield surface for kinematic strain-hardening material	7
3.6	Strain hardening softening curve for strain material	7
3.7	Flow chat of ABAQUS modeling and analysis steps	15
3.8	Eight-noded serendipity element	18
3.9	Warping in plate section	19
3.10	Elasto-plastic strain hardening behavior for the uniaxial case	27
4.1	Load diagram and plan view of plate	31
4.2	Convergence curve for different mesh size	32
4.3	Constitutive Relation for Perfectly plastic material	33
4.3	Stress flow pattern for plate having perfectly plastic properties	34
4.4	Load vs deflection curve for plate of 0.01m thickness in ABAQUS and MATLAB	34
4.5	Stress flow pattern for plate having perfectly plastic properties	35
4.6	Load vs deflection curve for plate of 0.02m thickness in	

	ABAQUS and MATLAB	38
4.7	Load vs deflection curve for plate of 0.03m thickness in ABAQUS and MATLAB	39
4.8	Comparison curve between load and deflection of plate of varying thickness in ABAQUS and MATLAB	39
4.9	Stress flow on plate having two opposite sides simply supported and other two sides free	40
4.10	Stress flow on plate having two opposite sides fixed and other two sides free	41
4.11	Stress flow on plate having three sides fixed and other side free	41
4.12	Stress flow on plate having all sides simply supported	42
4.13	Stress flow on plate having all sides fixed	42
4.14	Comparison curve between load and deflection values for different boundary conditions	43
4.15	Constitutive Relation for Strain hardening material	44
4.16	Stress flow pattern for plate having strain hardening property (ABAQUS)	45
4.17	Stress flow in strain hardening material from MATLAB	46
4.18	Load vs deflection curves for perfectly plastic and Strain hardening materials	47

LIST OF ABBREVIATIONS

a,b	: Length and width of plate
t	: Thickness of plate
[K]	: Stiffness matrix
[P]	: Applied load vector
[d]	: Deflection vector
[δ]	: Vector of unkown
ψ (δ)	: Residual load vector
ξ-η	: Natural co-ordinates
x_i, y_i	: Co-ordinates of ith node
N	: Shape functions
θx, θy	: Rotation in X and Y direction
ϕ_x, ϕ_y	: Shear deformation in X and Y direction
w	: Deflection in Z direction
ϵ	: Total Strain
σ	: Stress
ϵ_f , ϵ_s	: Strain in flexure and shear respectively
ϵ_x, ϵ_y	: Strain in X and Y direction
Υ_{xy},Υ_{xz},Υ_{yz}	: Components of shear strain
σ_f, σ_s	: Stress in flexure and shear plane

σ_x, σ_y	:	Stress in X and Y directions
τ_{xy},τ_{yz},τ_{zx}	:	Shear stress in Y, Z and X planes
M_x, M_y,M_{xy}	:	Moment components in XY plane
Q_x, Q_y	:	Shear force in X and Y directions
$\delta\epsilon_f$, $\delta\epsilon_s$	:	Incremental strain in flexure and shear
$\delta\epsilon_x$, $\delta\epsilon_y$	:	Incremental strain in X and Y directions
$\delta\gamma_{xy}$, $\delta\gamma_{xz}$, $\delta\gamma_{yz}$	:	Incremental shear strain
u	:	Nodal displacement
D	:	Rigidity matrix
D_f, D_s	:	Rigidity matrix in flexure and shear
E	:	Young's modulus
ν	:	Poison's ratio
χ	:	Hardening parameter
J_2'	:	Second deviatoric stress invariants
σ_1,σ_2, σ_3	:	Principal stresses
F	:	Plastic potential
$d\lambda$	:	Plastic multiplier

CHAPTER 1

INTRODUCTION

Chapter 1

1. Introduction

Plates and shells are very important parts of several engineering structures. Analysis and design of these origins are therefore always of interest to the scientific and engineering community. Accurate and conventional valuations of the maximum load the structure can carry, along with the equilibrium path followed in elastic and inelastic range are of paramount importance to understand realistic behavior of structures.

Elastic behavior of plates and shells have been very closely studied, mostly by using of the finite element method. On the other hand inelastic analysis, especially dealing with material nonlinearity, has received just handful attention from the researchers. The elasto-plastic behavior of structural elements are modelled using mathematical theory of plasticity and this includes analysis of flow of plastic deformations in the regions where the yield criteria is fulfilled.

For nonlinear analysis many commercial software's are available, such as ANSYS, ABAQUS, etc. All these software's are not tailor made applications which can work automatically on just feeding simple input data. An acceptable analysis of any structure by using these commercial software, and its correctness totally depends on the input values, especially when the material properties used.

1.1 Non-Linearity

Non-linear structural problems include the variation of stiffness of the structure with the change in deformation and the materials stress-strain behavior. Generally all physical structures exhibit non-linear behavior. Linear analysis is a convenient approximation that is often adequate for design purposes. It is obviously inadequate for many structural simulations including manufacturing processes, such as forging or stamping; crash analyses; and analyses of rubber components, such as tires or engine mounts. Since response of the structure to an external applied load is not linear, the load versus deflection curve will not be linear. The force and displacement relation for a spring with non-linear stiffening response is shown below.

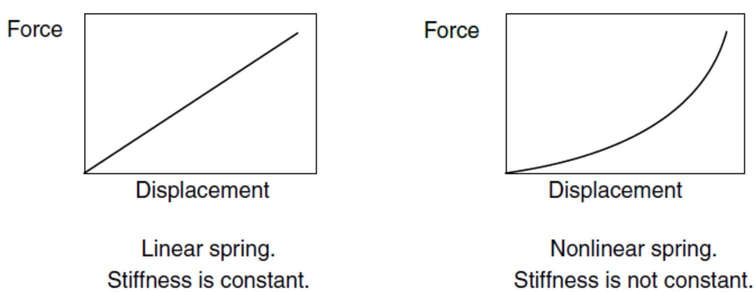


Figure 1: linear and nonlinear spring characteristics

As response nonlinear system is not linear function of magnitude of applied load, the stiffness cannot be directly calculated merely dividing load with deflection. Also it is not possible to create solutions for different load cases by superposition.

Types of Nonlinearity:

Before discussing the numerical methods, the sources of nonlinearity have been penned below. There are three types of nonlinearities.

1. Geometric nonlinearity
2. Material nonlinearity
3. Boundary nonlinearity

Geometric Nonlinearity:

This type of nonlinearity arises when large deflection affects the response of the structure.

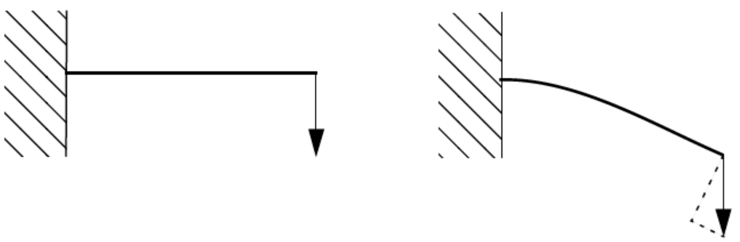


Figure 1.2: Large deflection of cantilever beam

Geometric nonlinearity can be three types:

a) Large displacement and small strain behavior: this deal with the smallness of one of the global coordinates of a body subjected to load.
b) Large displacement and large strain behavior**:** when changes to all the global coordinates of the body are comparable, the stress distribution in any direction cannot be neglected.
c) Linear stability analysis**:** if due to external load the body is on the verge of stable equilibrium and any further load will cause an unstable equilibrium in the system, the behavior of system can be considered as a linear function of applied load.

Material nonlinearity:

Material nonlinearity is caused due to the nonlinear relationship between stress and strain beyond the elastic limit. Beyond this limit some portion of the member will start yielding and based on materials, nonlinear constitutive relation start to respond in-elastically. This causes changes in the stiffness of the member which depends on the material behavior called Elasto-plastic behavior.

The present work involved only elasto-plastic behavior. An increase of the yield stress is referred to as *hardening* and its decrease is called *softening*. Typically, many materials initially harden and later soften as shown in Fig.1.4. A plot of a stress - strain curve defines material behavior.

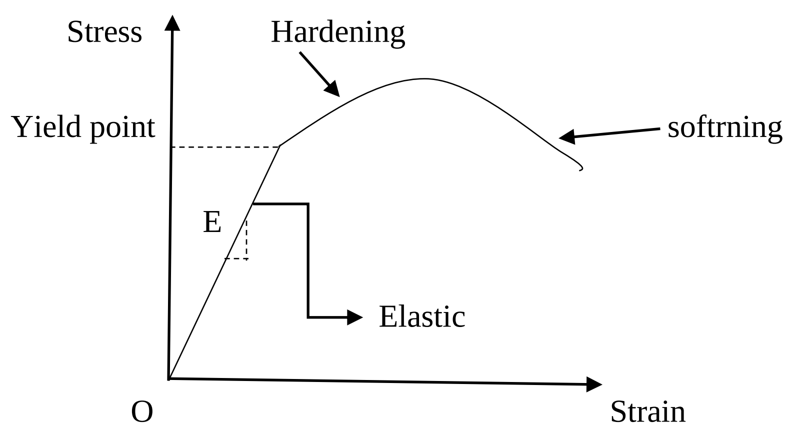


Figure 1.3: Strain Hardening behavior

Based on stress-strain diagram material behavior can be classified as

1. Perfectly Plastic
2. Elasto-plastic Strain Hardening
3. Elasto-plastic Strain softening

1. Perfectly plastic: It is the property of material for which the stress, strain curve of the material becomes parallel to strain axis, i.e. there is a large increase in strain for invariable yield stress value.

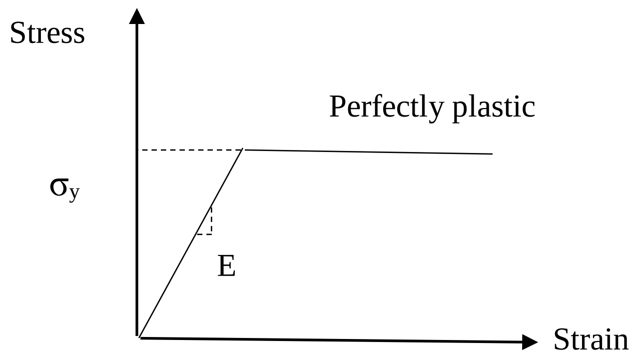


Figure 1.4: Perfectly plastic behavior

2. Elasto-plastic strain hardening: Materials exhibit a characteristic called work or strain hardening, which is strengthening of metal by plastic deformation. The yield surface for such materials will in general increase in size with further straining. It can also be classified into two types.

(i) Isotropic hardening (ii) Kinematic hardening

Isotropic hardening: It is characterized by the expanding yield surface of same shape with increasing stress (ref.Fig.1.5).

Kinematic Hardening: It is characterized by the yield surface of same shape and size translating in stress space (ref.Fig.1.6).

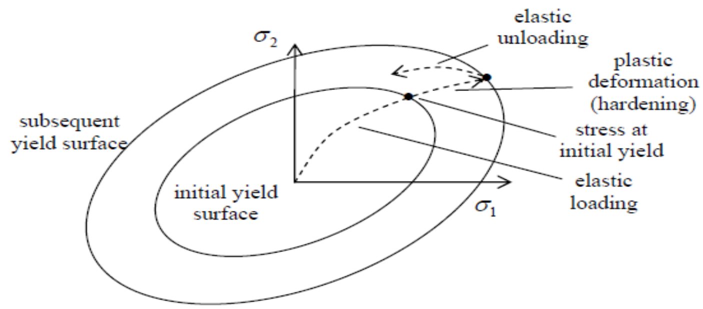


Figure 1.5: Isotropic hardening behavior

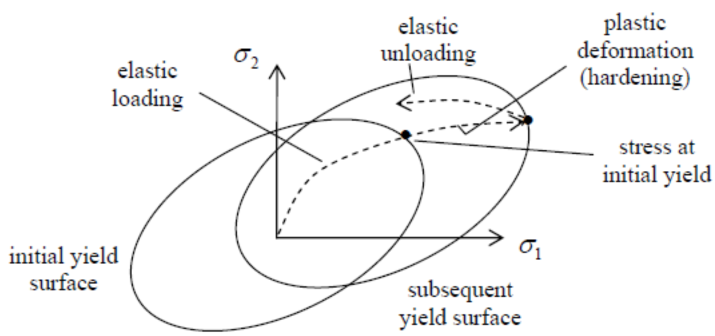


Figure 1.6: Kinematic hardening behavior

1.2 Objective of the Study

The aim of the present work is to perform static analysis of Mindlin plate involving only material nonlinearity incorporating isotropic hardening behavior by using ABAQUS software. The work involve modeling a simply supported plate in ABAQUS and analyzing it for two cases of material behavior, perfectly plastic and linear strain hardening (bilinear) behavior. The iterative method based on modified Newton Raphson's method was adopted for analysis. The load was applied in increments with a fixed load factor or time step. ABAQUS's quadrilateral shell element S8R was adopted with through the thickness stress integration (three point integration) points and Von Mises yield criteria. The effect of thickness and different boundary conditions on load carrying capacity, load deflection and spread or flows of plastic deformations are studied. The ABAQUS results were compared with reference solutions by Owen and Hinton (1980) and compared with results obtained from FEM based numerical analysis.

CHAPTER 2

REVIEW OF LITERATURE

Chapter 2

2. Literature review

The Reissiner-Mindlin plate theory is commonly used to analyze the bending behavior of elastic plate subjected to transverse force.

Owen D. R. J. and Hinton E. (1980), this book has presented and demonstrated the use of finite element based method for solution of problems involving plasticity. The detailed theory and algorithm in the form of modular coding written in FORTRAN is also given. Problems related to linear and nonlinear response of Mindlin and Kirchoff plates have been elaborately studied.

Talja A. and Pekka S. (1992**)** performed nonlinear analysis of two different cold formed high strength steel (HSS) beams, shorter one for were studied for bending resistance across the cross-section and longer beam for the lateral buckling. Both material and geometric nonlinearities were studied using FEM analysis. They used shell elements to model the beams. Strength calculation of beams showed that materials modeled with no imperfection can have 12 percent of accuracy in strength prediction.

Hui Shen Shen (2000) performed a nonlinear analysis for a simply supported rectangular moderately thick plate subjected to transverse central load resting on elastic supports. He considered first order shear deformation effect and the formulas were based on Reissner-Mindlin plate theory. Various numerical examples are solved and the performance of thick plate influenced by various factors like foundation stiffness, plate loaded area, aspect ratio of plate, transverse shear deformation has been analyzed.

WoelkePawel (2005) has presented a computational model for finite element, damaging and elasto-plastic analysis of homogeneous and isotropic shell. Considering the non-layered approach and updated Lagrangian method are used to describe the small strain geometric non-linearity. Though multi layered approach in composite shell gives more accurate results, in isotropic homogenous shells this becomes more complex, hence non layered approach is more realistic.

Szwaja Nicolas (2012) has performed elastic and elasto-plastic analysis of plates subjected to several boundary conditions and load using FEA ABAQUS. Here it can be concluded that by placing different conditions on a simple geometry like cavity and pores analysis, it is simpler to understand how FEA ABAQUS analyzes these models and how the strength of structure is related to changing conditions.

Fallah N. et al. (2014) developed a finite volume based analysis to study the elastoplastic bending behavior of plates. They applied Reissiner-Mindlin plate theory, performing layer-wise approach and two iteration methods. Their results of studies are compared with available reference from which it can be concluded that the developed finite volume based analysis has significant capabilities for solid mechanics analysis.

CHAPTER 3

THEORY AND FORMULATION

Chapter 3

3. Theory and formulation

3.1ABAQUS Modeling and analysis

In ABAQUS modeling and analysis include following three steps:

1. Preprocessing
2. Simulation
3. Postprocessing

3.1.1 Preprocessing

It is the initial step to analyze the physical problem. In this step model of the physical problem is defined and a ABAQUS input file (job.inp) in generated. Basic key points like material properties, element type, boundary condition, load, contact, mesh are defined here.

3.1.2 Simulation

The simulation is normally run as a background process. In this step already generated ABAQUS input file solves the numerical problem defined in the model. For example, output from a stress analysis, problem includes displacement and stress values and are stored in binary files in simulation which are further to be used in postprocessing. The output file is generated as job. odb.

During simulation ABAQUS uses Newton Raphson method to solve the non-linear type problems. Unlike linear analysis, load application to the system is incremental in non-linear case. ABAQUS breaks the simulation stage into number of *load increments* and at the end of each load increment it finds an approximate equilibrium configuration. Sometimes ABAQUS takes a number of iterations to find an acceptable solution depends on tolerance specified, for a particular load increment. Finally the cumulative summation of all load incremental responses is the approximate solution to that non-linear problem. This way ABAQUS uses both incremental and iterative methods to solve non-linear problems. There are three phases in simulation stage

- Analysis step
- Load increment
- Iteration

Analysis step which generally consists of loading option, output request. Output request describes the values of required parameters like displacement, stress, strain, reaction force, bending moment etc.

In increment step, first load increment is to be defined by the user and the subsequent increments will be chosen by ABAQUS automatically.

Iteration continues till ABAQUS optimize the residual forces to the given tolerance value.Hence, after each load increment the structure satisfies the equilibrium conditions and corresponding output request values are to be written to the output database file.

3.1.3 Postprocessing

Once the simulation is over, the calculated variables like stresses, displacements, strain, reaction forces etc. can be displayed through Visualization module of ABAQUS. The visualization module has a variety of options to display the results such as animation, color contour plots, deformed shape plots and *X*-*Y* plots.

In our work, it is required to find out deflections, plastic strain, stresses and yield stress values at specified nodes. All these values can be obtained from the visualization module of ABAQUS.

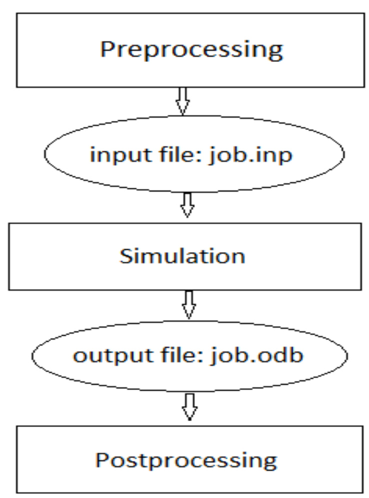


Figure 3.1: Flow-chat of ABAQUS modeling and analysis

3.2.3 Element Type

The correct choice of element for a particular simulation is very important to meet the desired accuracy in the result. The ABAQUS/Standard solid element can be classified into two categories.

a) First order (linear) interpolation element

b) Second order (quadratic) interpolation element

The first order element type is used for the two dimensional analysis and second order element type is used for three dimensional analysis. While selecting element, ABAQUS gives a number of choices like linear, quadratic, brick etc. In the present study 8 noded quadratic shell element S8R is adopted where R represents reduced integration type, S stands for shell element having 8 nodes in which each node is assigned with 6 degrees of freedom. The present study involves very large mesh distortions and large strain analysis.

3.2 Formulation for Finite Element Method

3.3.1 Equilibrium equations

$$[K]\,\delta + P = \psi(\delta) \neq 0 \tag{1}$$

Where [K] is assembled stiffness matrix

P is vector of applied load

δ is vector of basic unknown i.e. defections d

$\psi(\delta)$ is vector of residual force.

If the coefficients of the matrix K depend on the unknowns δ or their derivatives, the problem clearly becomes nonlinear. In this case, direct solution of equation system (1) is generally impossible and an iterative scheme must be adopted. For nonlinear situations, in which the stiffness depends on the degree of displacement in some manner, K is equal to the local gradient of the force-displacement relationship of the structure at any point and is termed the tangential stiffness. The analysis of such problems must proceed in an incremental manner since the solution at any stage may not only depend on the current displacements of the structure, but also on the previous loading history. In present study Newton-Raphson technique following the tangential stiffness method is adopted for nonlinear analysis of Mindlin plate.

3.3.2 Discretization:

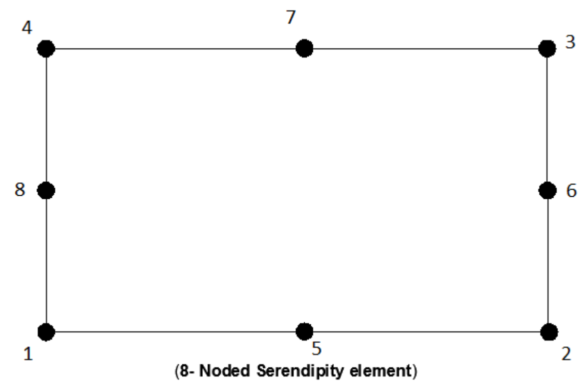
The arbitrary shape of the whole plate is mapped into a Master Plate of square region [-1, +1] in the ξ-η plane with the help of the relationship given by

$$x = \sum_{i=1}^{8} N_i(\xi,\eta)x_i \tag{2}$$

$$y = \sum_{i=1}^{8} N_i(\xi,\eta)y_i \tag{3}$$

where (x_i, y_i) are the coordinates of the i^{th} node on the boundary of the plate in the x-y plane and

$N_i(\xi,\eta)$ are the corresponding cubic Serendipity shape functions presented below.



8-noded Serendipity element

$N_1 = {}^{1}/_{4}\ (\eta - 1)\ (1-\xi)\ (\eta + \xi + 1)$

$N_2 = {}^{1}/_{2}\ (1 - \eta)\ (1-\xi^{2})$

$N_3 = {}^{1}/_{4}\ (\eta - 1)\ (1-\xi)\ (\eta - \xi + 1)$

$N_4 = {}^{1}/_{2}\ (1 - \eta^{2})\ (1 + \xi)$

$N_5 = {}^{1}/_{4}\ (1 + \eta)\ (1 + \xi)\ (\eta + \xi - 1)$

$N_6 = {}^{1}/_{2}\ (1 + \eta)\ (1-\xi^{2})$

$N_7 = {}^{1}/_{4}\ (1 + \eta)\ (1 - \xi)\ (\eta - \xi - 1)$

$N_8 = {}^{1}/_{2}\ (1 - \eta^{2})\ (1 - \xi)$

$$[N] = [N_1\ N_2\ N_3\ N_4\ N_5\ N_6\ N_7\ N_8] \tag{4}$$

3.3.3 Plate element formulation:

The displacement field at any point within the element is given by

$$\{U\} = \begin{bmatrix} u - z\,\theta_x(x,y) \\ u - z\,\theta y(x,y) \\ w(x,y) \end{bmatrix} \tag{5}$$

Owing to the shear deformations, certain warping in the section occurs as shown in Fig. 3.3. However, considering the rotations θx and θy as the average and linear variation along the thickness of the plate, the angles ϕ_x and ϕ_y denoting the average shear deformation in and x-y

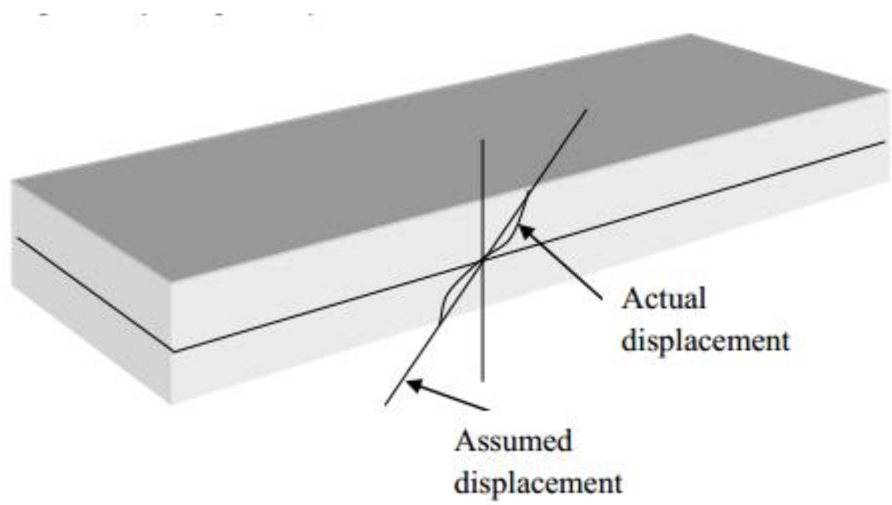


Figure 3.3: warping in plate section

$$\begin{Bmatrix} \theta_x \\ \theta_y \end{Bmatrix} = \begin{Bmatrix} \frac{\partial w}{\partial x} + \varphi_x \\ \frac{\partial w}{\partial y} + \varphi_y \end{Bmatrix} \tag{6}$$

The plate strains are described in terms of middle surface displacements i. e. x-y plane coincides with the middle surface .The strain matrix is given by

$$\{\epsilon\} = \begin{Bmatrix} \epsilon_f \\ \epsilon_s \end{Bmatrix} = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \Upsilon_{xy} \\ \Upsilon_{xz} \\ \Upsilon_{yz} \end{Bmatrix} \tag{7}$$

And stress matrix is given by

$$\{\sigma\} = \begin{Bmatrix} \sigma_f \\ \sigma_s \end{Bmatrix} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} \tag{8}$$

For non-layer approach

We interpret

$$\left[\sigma_f\right] = [\, M_x \quad M_y \quad M_{xy}]^{\mathrm{T}} \tag{9}$$

and
$$[\sigma_s = \ [\, Q_x \quad Q_y]^{\mathrm{T}} \tag{10}$$

Since iterative method is used for analysis, the corresponding relations in incremental form can be written as

$$\{\delta\epsilon\} = \begin{Bmatrix} \delta\epsilon_f \\ \delta\epsilon_s \end{Bmatrix} = \begin{Bmatrix} \delta\epsilon_x \\ \delta\epsilon_y \\ \delta\gamma_{xy} \\ \delta\gamma_{xz} \\ \delta\gamma_{yz} \end{Bmatrix} \tag{11}$$

$$\delta\epsilon_f = z\left[-\frac{\partial\delta\theta_x}{\partial x} \quad -\frac{\partial\delta\theta_y}{\partial y} \quad -(\frac{\partial\delta\theta_y}{\partial x}+\frac{\partial\delta\theta_x}{\partial y})\right]^{\mathrm{T}} \tag{12}$$

$$\delta\epsilon_s = \left[\frac{\partial\delta w}{\partial x}-\delta\theta_x, \quad \frac{\partial\delta w}{\partial y}-\delta\theta_y\right]^{\mathrm{T}} \tag{13}$$

3.3.4 Strain displacement relationship:

For an isotropic material the displacement can be written as

$$\mathrm{U}=\sum_{i=1}^{8} N_i(\xi,\eta)u_i \tag{14}$$

Where u_i is nodal displacement vector at i^{th} node may be represented as

$$u_i = [w_i, \theta_{xi}, \theta_{yi}]^{\mathrm{T}} \tag{15}$$

$$\mathrm{U} = [w, \theta_x, \theta_y]^{\mathrm{T}} \tag{16}$$

The **flexural strain –displacement equation** in incremental form is given as

$$\delta\epsilon_f = \sum_{i=1}^{8} B_{fi}\delta u_i \tag{17}$$

Where

$$B_{fi} = \begin{bmatrix} 0 & -\frac{\partial N_i}{\partial x} & 0 \\ 0 & 0 & -\frac{\partial N_i}{\partial y} \\ 0 & -\frac{\partial N_i}{\partial y} & -\frac{\partial N_i}{\partial x} \end{bmatrix} \tag{18}$$

The incremental shear strain displacement equation is

$$\delta\epsilon_s = \sum_{i=1}^{8} B_{si}\delta u_i \tag{19}$$

Where $$B_{si} = \begin{bmatrix} \frac{\partial N_i}{\partial x} & -N_i & 0 \\ \frac{\partial N_i}{\partial y} & 0 & -N_i \end{bmatrix} \tag{20}$$

3.3.5 Virtual work Equation

Giving a virtual displacement δu to the system the virtual work statement may be written as

$$\sum_{i=1}^{n} [\delta u_i]^T \left\{ \int A \int_{-t/2}^{t/2} \left[B_{fi}\right]^T \sigma'_f z + [B_{si}]^T \sigma_s' z - [N_i]^T q \right\} dz\, dA = 0 \tag{21}$$

or $$\sum_{i=1}^{n} \psi i(u) = 0$$

where ψ_i is residual force vector at i^{th} node.

Since equation (21) must be true for any set of virtual displacements we get (for layered model)

$$\left\{ \int A \int_{-t/2}^{t/2} \left[B_{fi}\right]^T \sigma'_f z + [B_{si}]^T \sigma'_s z - [N_i]^T q \right\} dz\, dA = 0 \tag{22}$$

For nonlayer model

$$\int_A \left[\left[B_{fi}\right]^T \sigma_f + [B_{si}]^T \sigma_s - [N_i]^T q \right] dA = 0 \tag{23}$$

$$\psi = [\psi_1, \psi_2, \psi_3, \ldots\ldots\ldots \psi_n]^T \tag{24}$$

Contribution to residual force vector is evaluated at element level and then assembled to for residual force vector ψ.

3.3.5Formulation in inelastic region

In this study material non linearity due to an elasto-plastic material response is considered and isotropic effects are included in the yielding behavior. To model elasto-plastic material behavior in inelastic region two conditions have to be met:

1. A yield criterion representing the stress level at which plastic flow commences must be postulated,

2. A relationship between stress and strain must be developed for post yielding behavior.

Before onset of yielding the relationship between stress and strain is given by

$$\sigma = D^*\varepsilon \tag{25}$$

D is rigidity matrix

$$D = \begin{bmatrix} D_f \\ D_s \end{bmatrix} \tag{26}$$

$$D_f = \frac{Et^3}{12(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{(1-\nu)}{2} \end{bmatrix}$$

$$D_s = \frac{Et}{2.4(1+\nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \tag{28}$$

For the isotropic material the yield criteria adopted is a generalization of the Von Mises law.

The Von Mises Yield Criterion:

In general form yield criterion is written as

$$F(\sigma, \chi) = f(\sigma) - Y(\chi) = 0 \tag{29}$$

where f is some function of the deviatoric stress invariants and Y is yield level which is function of hardening parameter χ.

Defining the effective stress $\boldsymbol{\sigma}$ for isotropic Von Mises material as

$$\boldsymbol{\sigma} = \sqrt{3}k \tag{30}$$

where

$$k = (J_2')^{1/2} \tag{31}$$

and J_2' is the second deviatoric stress invariants

$$J_2' = \frac{1}{6}\left[(\sigma_1 - \sigma_2)^2 + [(\sigma_2 - \sigma_3)^2 + [(\sigma_3 - \sigma_1)^2\right]$$

where σ_1,σ_2,σ_3 are principal stresses

$$= \frac{1}{2}\left[\sigma_x'^2 + \sigma_y'^2 + \sigma_z'^2\right] + \tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2 \tag{33}$$

3.3.6 Elasto-plastic stress strain relation

After initial yielding the material behavior will be partly elastic and partly plastic. During any increment of stress, the changes of strain are assumed to be divisible into elastic and plastic components, so that

$$d\varepsilon = d\varepsilon_e + d\varepsilon_p \tag{34}$$

The elastic strain increment is given by the incremental form of

$$d\varepsilon_e = [D]^{-1} d\sigma \tag{35}$$

and the plastic strain increment by the flow rule

$$d\varepsilon_p = d\lambda \frac{\partial Q}{\partial \chi} \tag{36}$$

where Q is defined as plastic potential and $d\lambda$ is a proportional constant called plastic multiplier.

The assumption $Q \equiv \boldsymbol{f}$ gives rise to an associated plasticity theory, in which case equation (36) represents the normality condition; since $\frac{\partial f}{\partial \sigma}$ is a vector directed normal to the yield surface in a stress space geometrical interpretation.

The differential form of eq. (29) is

$$dF = \frac{\partial F}{\partial \sigma} d\sigma + \frac{\partial F}{\partial \chi} d\chi = 0 \tag{37}$$

or

$$a^T d\sigma - A d\lambda = 0 \tag{38}$$

in which the flow vector $\mathbf{a^T}$ is define as

$$a^T = \frac{\partial F}{\partial \sigma} = \left[\frac{\partial F}{\partial \sigma x}, \frac{\partial F}{\partial \sigma y}, \frac{\partial F}{\partial \tau xy}, \frac{\partial F}{\partial \tau yz}, \frac{\partial F}{\partial \tau zx}\right] \tag{39}$$

Equation (37) & (38) can be reduced to get

$$A = -\frac{1}{d\lambda}\frac{\partial F}{\partial \chi}d\chi \tag{40}$$

Total incremental strain is

$$d\varepsilon = [D]^{-1}d\sigma + d\lambda\ \frac{\partial F}{\partial \chi} \tag{41}$$

Pre-multiplying both sides by a^T D and eliminating $a^T d\sigma$ by using eq. (40), we get $d\lambda$ to be

$$d\lambda = \frac{1}{[A + a^T D\ a]} a^T\ D^T\ a\ d\varepsilon \tag{42}$$

Manipulation of equation (34) to equation (42) will give elasto-plastic incremental stress strain relationship

$$D\sigma = D_{ep} d\varepsilon \tag{43}$$

Where $$D_{ep} = D - \frac{D\ a\ a^T\ D}{[A + a^T D\ a]} \tag{44}$$

The hardening parameter A can be deduced from uniaxial conditions as

$$A = H' = \frac{\partial \sigma}{\partial \epsilon_p} \tag{45}$$

Thus A is obtained to be the local slope of the uniaxial stress/plastic strain curve and can be determined experimentally from Fig.

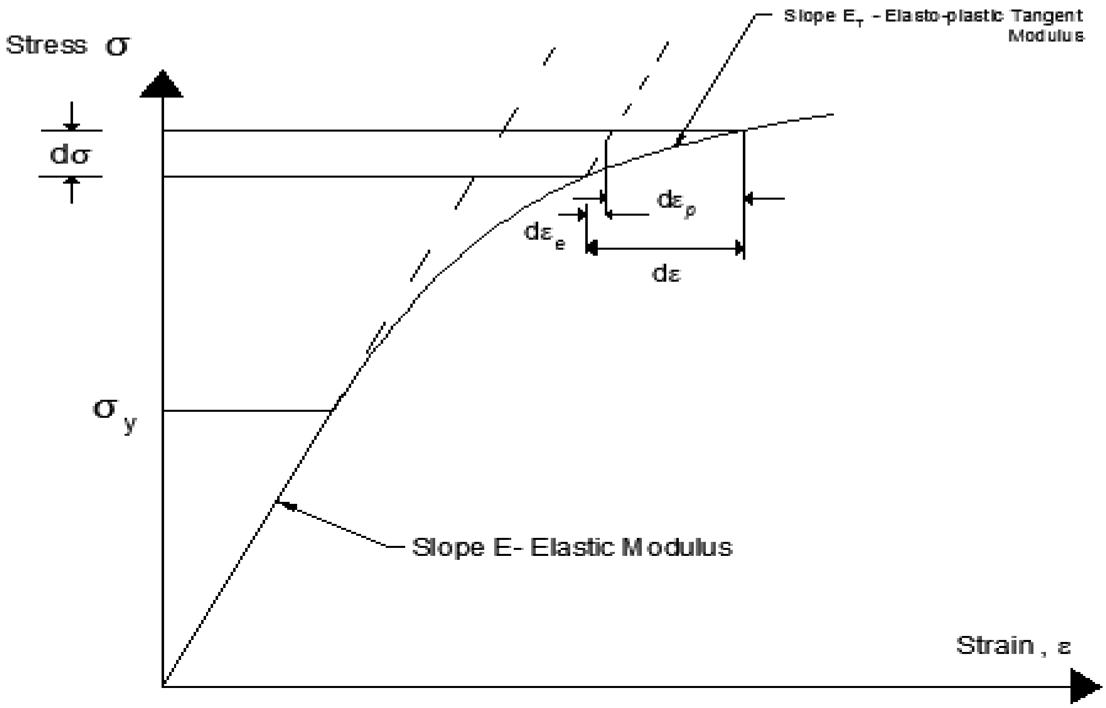


Fig. 3.4: Elasto-plastic strain hardening behavior for the uniaxial case

$$A = H' = \frac{E_T}{1 - {E_T}/{E}} \tag{46}$$

The incremental stress-strain resultant relationship is given as

$$\begin{bmatrix} d\sigma_f \\ d\sigma_s \end{bmatrix} = \begin{bmatrix} (D_{ep})_f & 0 \\ 0 & D_s \end{bmatrix} \begin{bmatrix} d\varepsilon_f \\ d\varepsilon_s \end{bmatrix} \tag{47}$$

For Mindlin plate, yield function F is assumed to be function of σ_f, the direct stresses associated with flexure only hence D_s always remain elastic.

3.3.7 Tangential Stiffness matrix:

From equation (22),the tangential stiffness matrix can be written as

$$K_T = \int_A \left[[B_f]^T (D_{ep})_f \, B_f + [B_s]^T D_s B_s \right] d_A \tag{48}$$

CHAPTER 4

RESULT AND DISCUSSION

4.1 Methodology

1. A convergence study was performed to fix the mesh size for analysis in ABAQUS.
2. To check the accuracy of element selected, time step, analysis method elastic analysis was performed using ABAQUS and results were compared with exact solution.
3. The plate was analysed considering perfectly plastic material behaviour. The results were validated with reference solutions by Owen and Hinton (1980).The results were also compared with results obtained from FEM based numerical solutions.
4. The effect of thickness and boundary conditions were studied.
5. The plate was analysed considering strain hardening material behaviour. The results were validated with numerically solved results.

4.2 Convergence study

The convergence study is carried out to determine the mesh size or the number of elements required for the Finite Element Analysis. A plate having all sides simply supported is subjected to a uniformly distributed load of magnitude 1 kN/m^2.

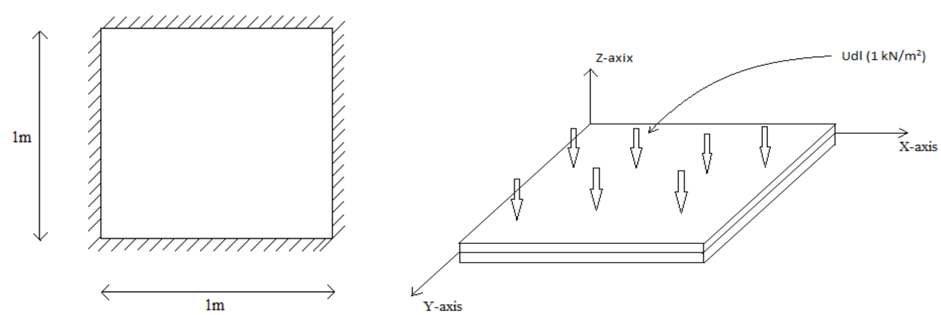


Figure 4.1: Load diagram and plan view of plate

Applying the udl in increment with a constant load factor 0.02, the deflection value at the midpoint on plate for different load increments had been calculated. For load factor 0.5, central deflection for different mesh size were tabulated in Table 4.1 below.

Table 4.1: Convergence study for central deflection

Deflection Value at midpoint	Mesh division						
	2×2	4×4	6×6	8×8	12×12	16×16	20×20
	950.914	2010.31	2031.15	2032.65	2032.80	2032.80	2032.80

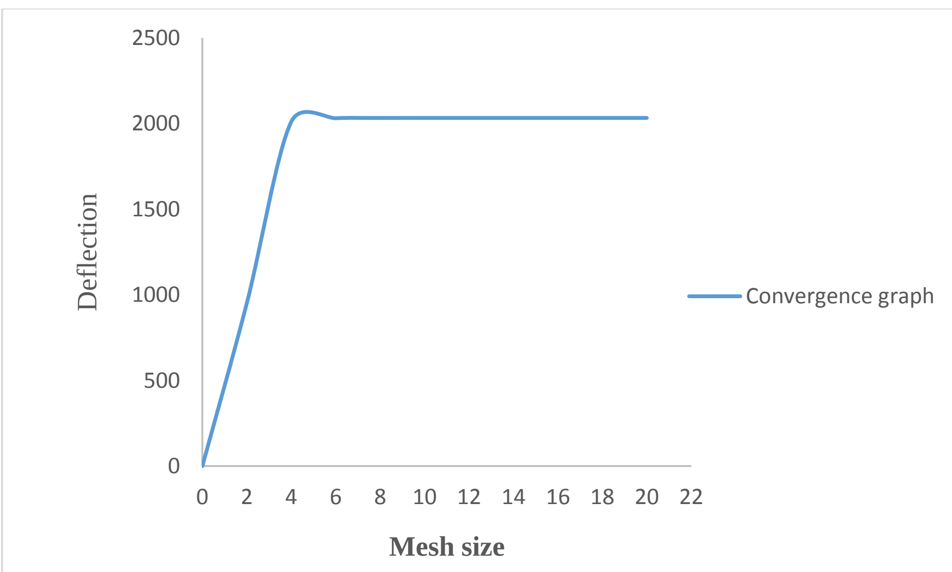


Fig 4.2: convergence curve

From the above table it can be concluded that the deflection value after taking mesh size (12×12) and onwards does not vary much i.e. results show good convergence for mesh division (12×12). Hence 12x12 mesh division is used for further study.

4.2 Problem Statement 1

A simply supported plate subjected to uniformly distributed load was analysed to predict first yield load, collapse load. The load–displacement response and plastic flow patterns were critically observed. Only material non-linearity were examined to allow for validation with the reference solution by Owen and Hinton (1980). The results were compared with the results obtained from FEM based analysis developed on the formulation given and written in MATLAB.

A simply supported square plate of dimension 1m × 1m, subjected to a uniformly distributed load of magnitude q = 1.5kN/m^2 was analyzed for linear, perfectly plastic and strain hardening behavior. Given

Youngs modulus E = 10.92kN/m^2, Poisson's ratio= v = 0.3

Yield stress σ_0 = 1600 N/mm^2. Thickness of plate = 0.01m.

4.2.1 Analysis of perfectly plastic material behavior:

The elasto-plastic analysis of plate was performed for perfectly plastic material. The constitutive relation of the material is shown in the Figure.

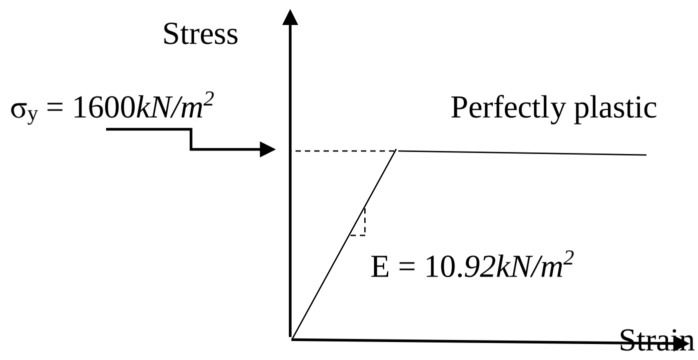


Fig 4.3: Constitutive Relation for Perfectly plastic material

In the analysis load is applied in increments with a load factor of 0.02. The iterative method is adopted with ABAQUS's quadrilateral shell element S8R with through the thickness stress integration (3) points and Von Mises yield criteria. The load deflection curve (Fig 4.4) obtained from numerical method (MATLAB results) and ABAQUS was presented in a non-dimensional form.

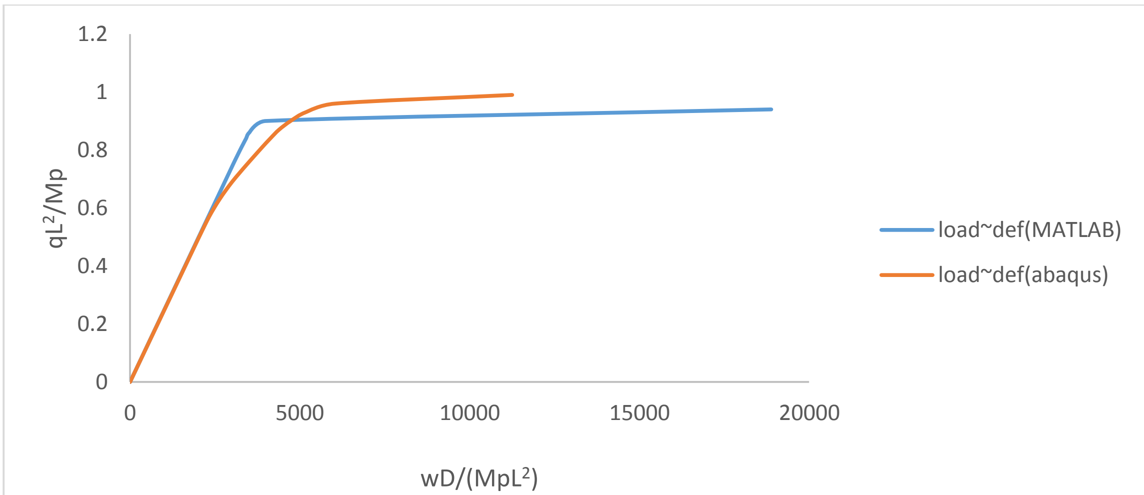


Fig 4.4: Load-deflection diagram for S/S plate of 0.01m thickness

From the graph the maximum deflection occurs at load 0.99kN/m^2 in ABAQUS and0.94kN/m^2 in MATLAB. The first yield occurs at load 0.81kN/m^2 and the collapse load is 0.99kN/m^2.

Comparison with reference data:

Central deflection is compared at load factor 0.856. ABAQUS result is compared with reference data by Owen and numerical result from MATLAB coding. Nonlinearity for transverse shear is not considered in FEM modeling by Owen and the analysis adopted is non-layered approach

whereas ABAQUS considered nonlinearity for flexure and transverse shear both. Hence there is difference in results. That shows the effect of residual transverse shear.

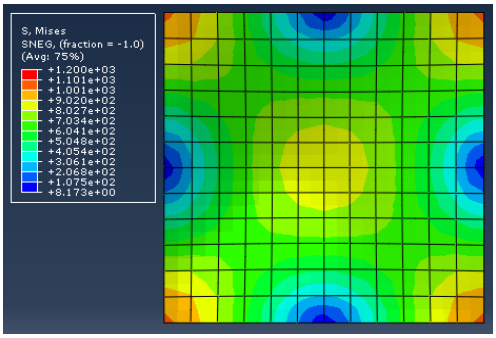
Table 4.2: Deflection value at center from ABAQUS, MATLAB and reference data

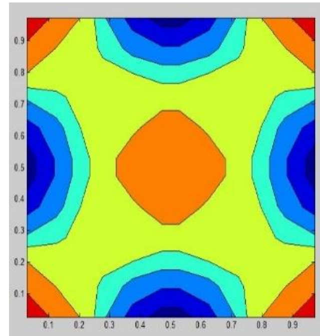
Present study (from ABAQUS)	Numerical result (from MATLAB)	Reference Value (from Owen and Hinton)
3739.923×10^{-3}	3494.82×10^{-3}	3496.31×10^{-3}

Plastic Flow pattern:

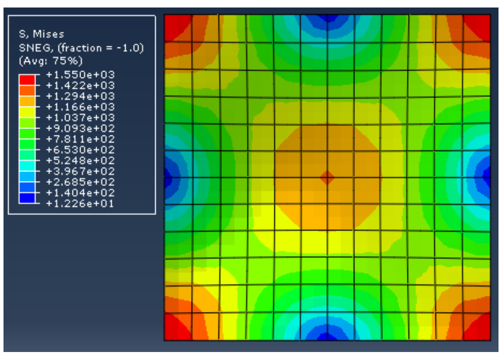
The main objective of the present work is to track the progressive plasticization of the cross sections. Since the stresses are calculated at different load steps, yielding of sections can be easily tracked which give an idea of plastic flow in the plate. The Figures give an idea of plastic flow in simply supported plate. The side by side figures represent stress contours obtained at different stages of loadings, from ABAQUS software and MATLAB coding.

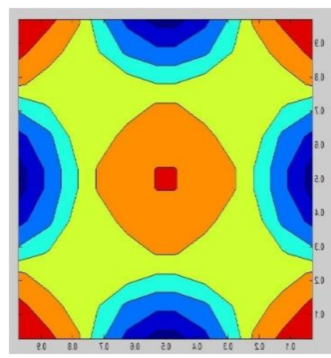
<u>ABAQUS</u> <u>MATLAB</u>



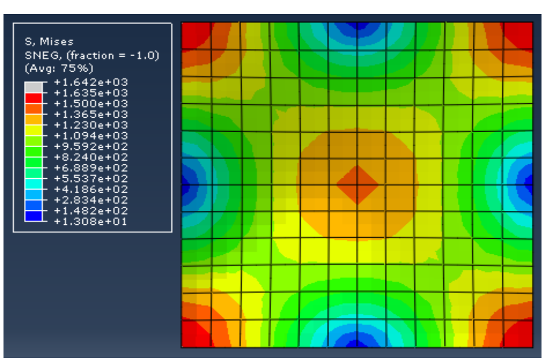


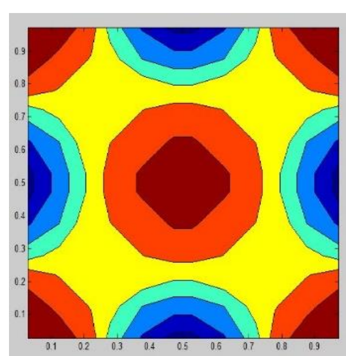
Stress Contour for load q=0.15kN/m^2 (L.F. =5)



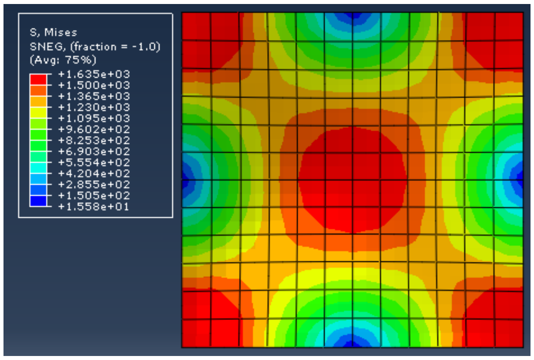


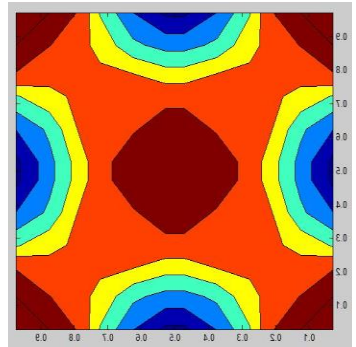
q = 0.45kN/m^2 (L.F. =15)



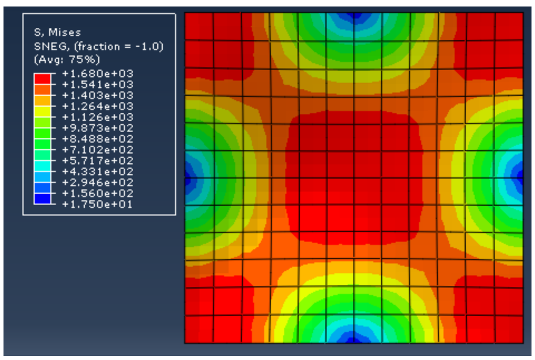


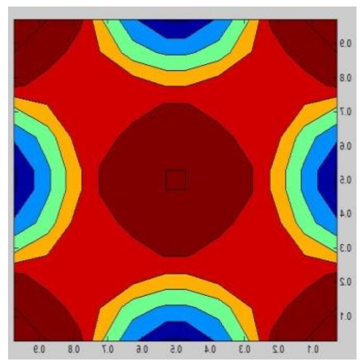
q = 0.48kN/m^2(L.F. =16)



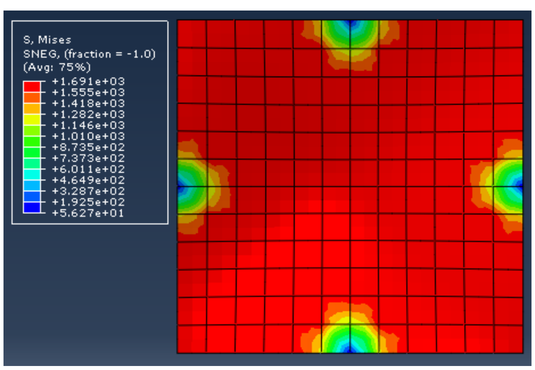


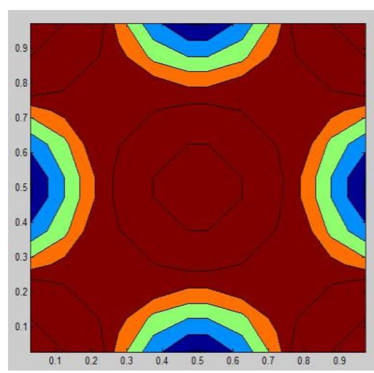
q = 0.60kN/m^2 (L.F. =20)





q = 0.75kN/m^2(L.F. =25)





q = 0.99kN/m^2 (L.F. =33) (q = 0.94kN/m^2)

Fig 4.5: stress flow pattern for plate having perfectly plastic properties

The plastic flow followed the yield line pattern for s/s slab. The initial yielding started at corner points and further spread justifying the well-known corner lever effect in a simply supported two way slab used in yield analysis of slab. This causes lifting of corners. Both analysis exhibited same phenomenon and same pattern of plastic flow. A comparison for yield and collapse loads are given in Table.

Effect of thickness:

The FEM formulation did not include non-linearity in transverse shear while ABAQUS analysis takes care of it. Hence to observe the effect of thickness on elasto-plastic behavior of simply supported plate further analysis were carried out by taking thickness as 0.02 and 0.03. This would gave an idea about the influence of the transverse shear on elasto-plastic behavior of plate. For 0.01m thickness this influence is very small. The effect were observed in terms of load-deflection curves, yield load and collapse load by using numerical method (MATLAB)and ABAQUS. The curves are shown in Fig and Fig. and Table gives a comparison between yield and collapse loads.

Table 4.3: Yield and collapse load for a square plate with different thickness

Thickness(m)	ABAQUS		MATLAB	
	Load (kN/m^2) at first yield	Load (kN/m^2) at collapse	Load (kN/m^2) at first yield	Load (kN/m^2) at collapse
0.01	0.54	0.99	0.54	0.94
0.02	2.57	4.00	2.57	3.80
0.03	5.40	8.80	5.40	7.92

Increasing the thickness yield load and collapse load were observed to increase.

Thickness (t) = 0.02 m

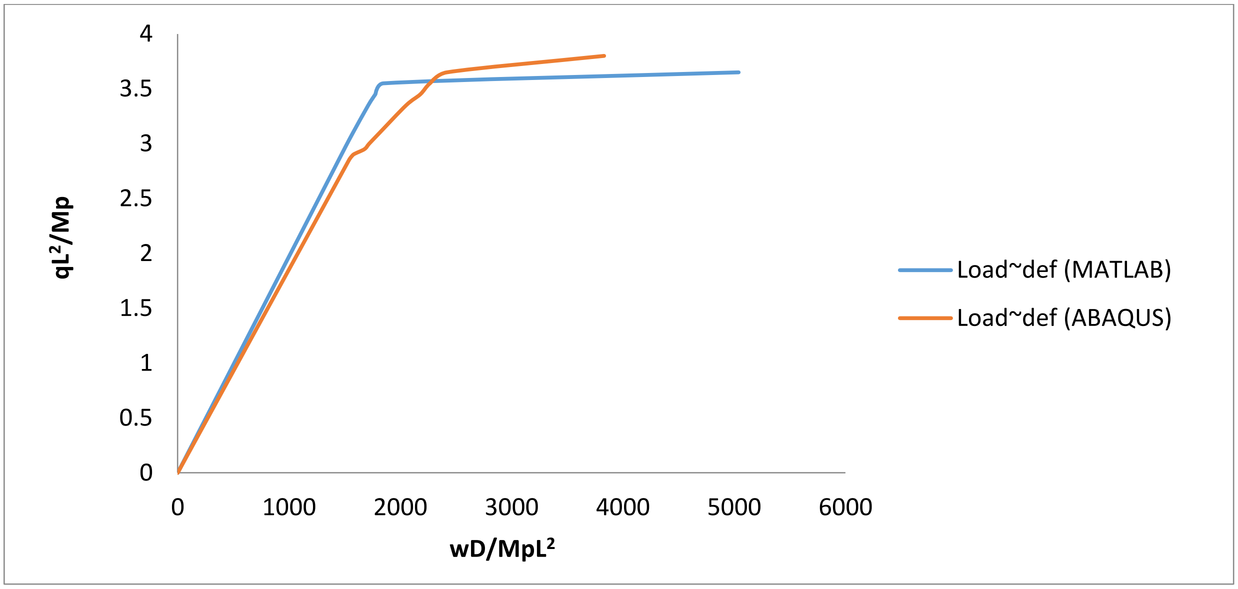


Fig 4.6: Load vs deflection curve for plate of 0.02m thickness

Thickness (t) = 0.03 m

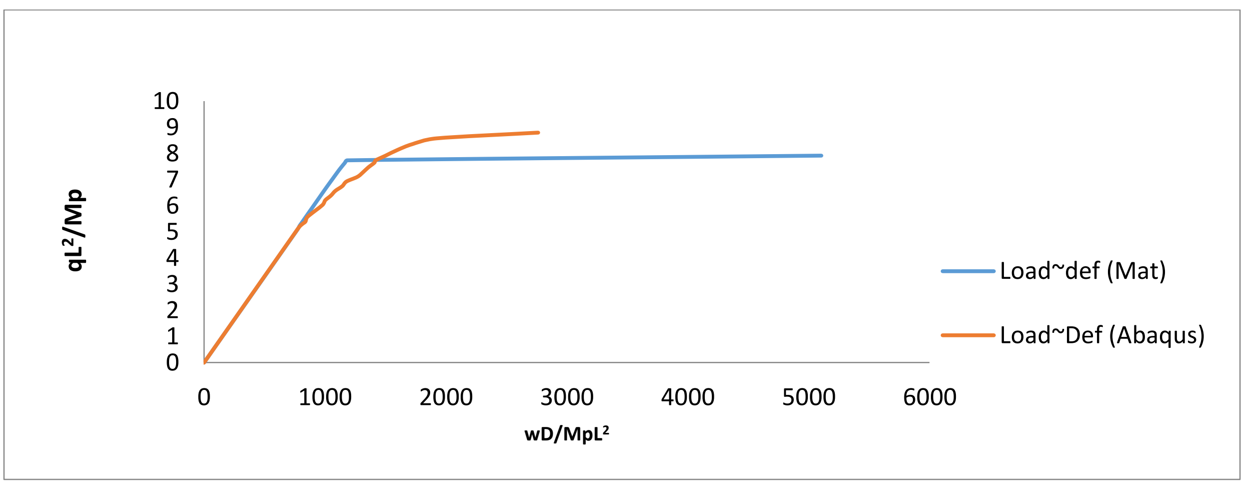


Fig 4.7: Load vs deflection curve for plate of 0.03m thickness

A comparison graph between yield load and collapse load at different thickness obtained from ABAQUS Software and MATLAB modeling is represented below.

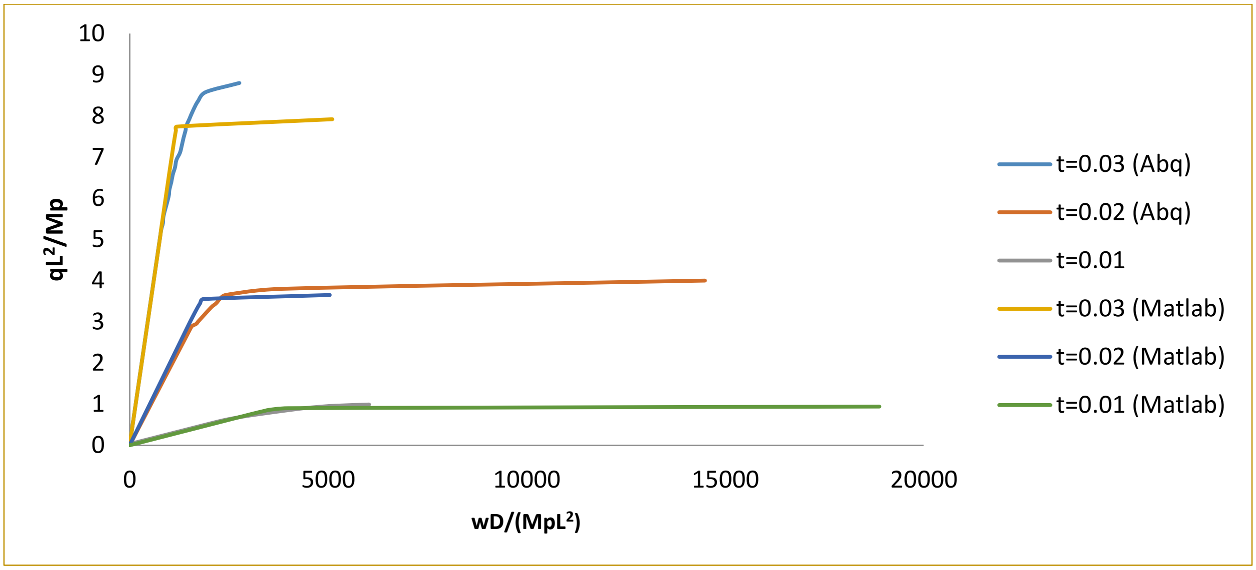


Fig 4.8: Load vs deflection curves for plate of varying thickness

Since ABAQUS considers nonlinearity in flexure and transverse shear both, a higher deflection value is observed as compared to numerical approach (MATLAB coding). Also the graph concludes a larger variation in value with increased thickness value.

<u>Effect of Boundary Conditions:</u>

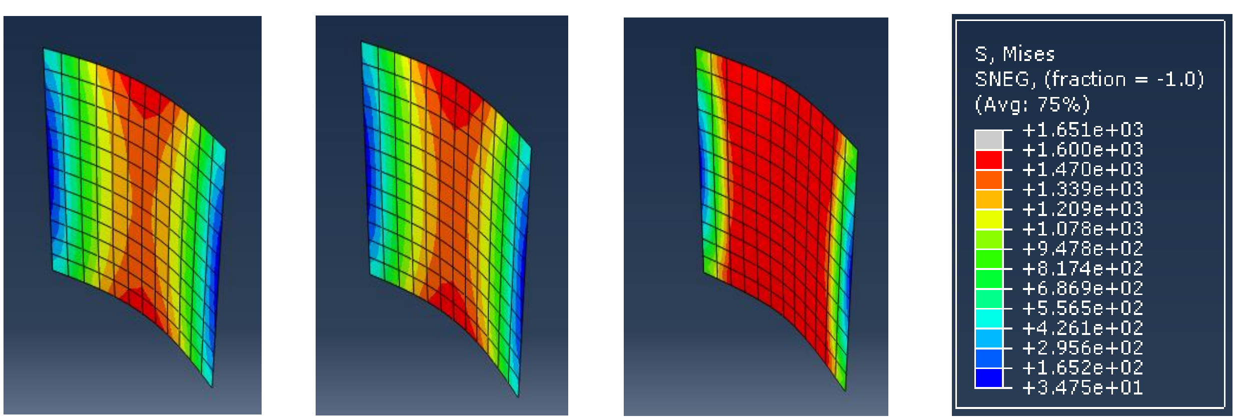
The plate of same dimensions was analyzed for different boundary conditions by ABAQUS software. The variation were observed in terms of yield load, collapse load and plastic flow. The comparison of yield and collapse loads is given in Table

Table 4.4: Yield and collapse load for a square plate with different boundary conditions

Boundary condition	Yield load (*kN/m²*)	Collapse load (*kN/m²*)
Two opposite sides simply supported and other two free	0.24	0.36
Two opposite sides fixed and other two free	0.60	0.78
All sides simply supported	0.72	0.99
Three sides fixed and one free	0.60	1.11
All sides fixed	1.62	3.42

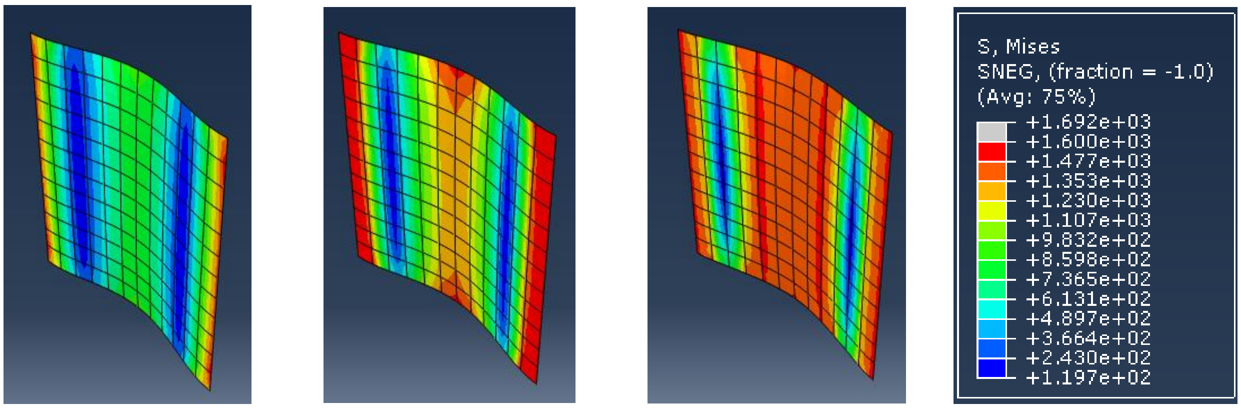
Plastic flow for different boundary condition was observed similar to yield line adopted in Yield line theory of slab.

<u>Fig.4.9 Plastic Flow for two opposite sides s/s and other two sides free</u>:



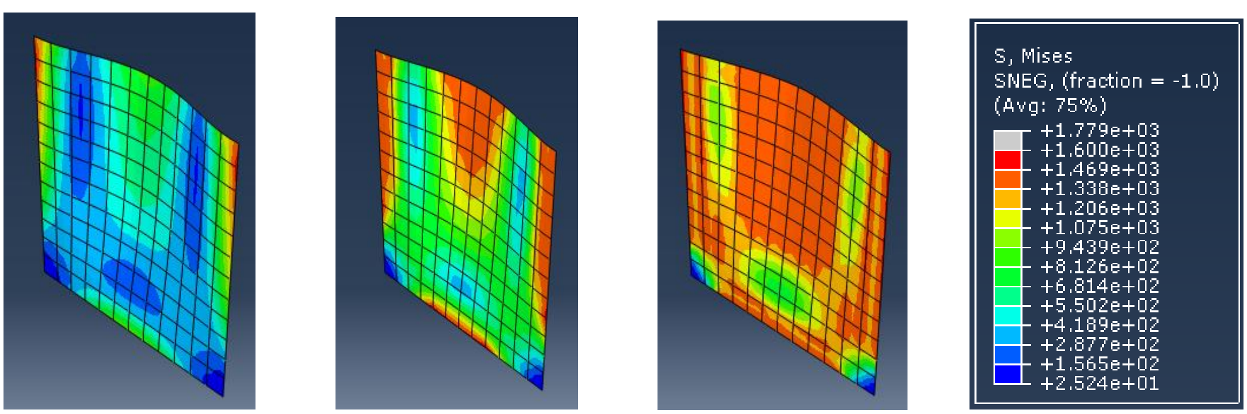
(q = 0.21*kN/m²*) (q = 0.24*kN/m²*) (q = 0.36*kN/m²*)

Figure 4.10 Plastic Flow for Two opposite sides fixed and other two free:



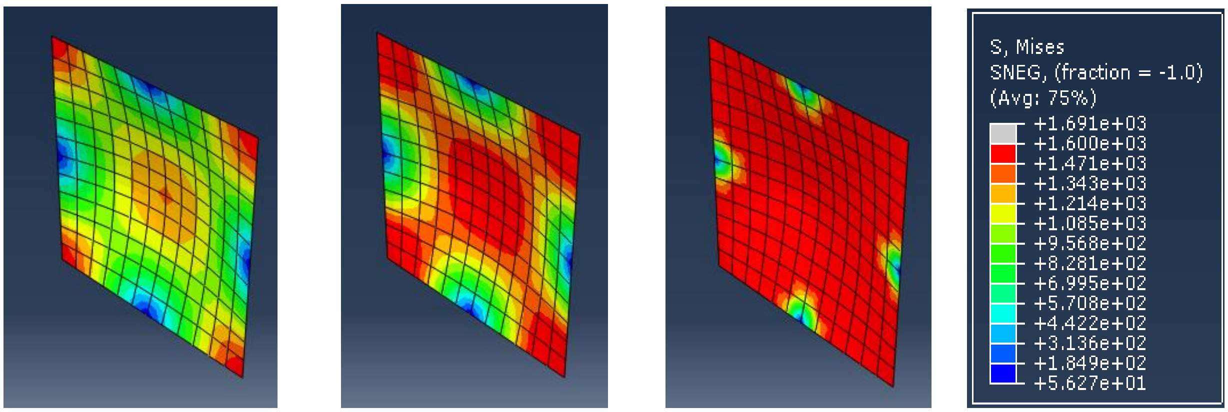
(q = 0.30kN/m^2) (q = 0.60kN/m^2) (q = 0.78kN/m^2)

Figure 4.11 Plastic Flow for Three sides fixed supported:



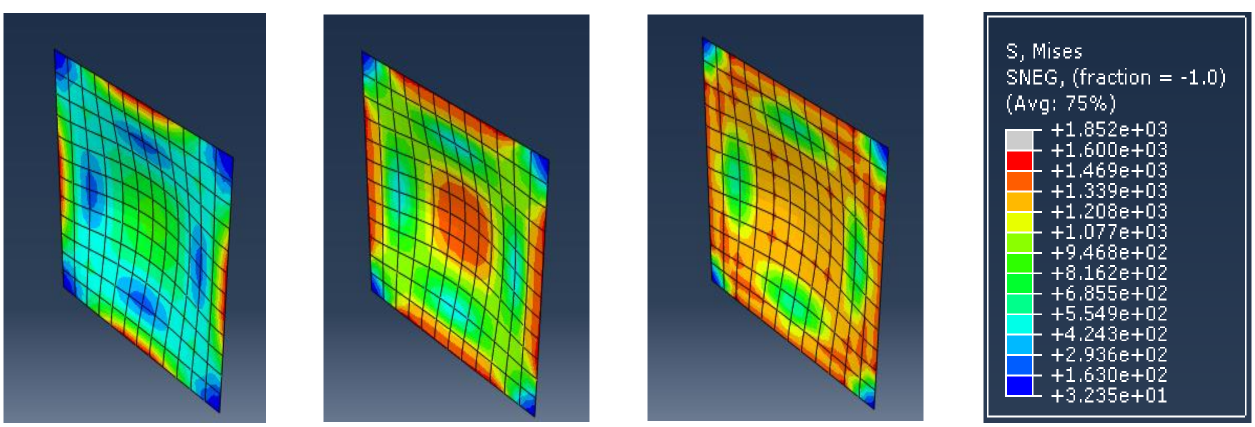
(q = 0.75kN/m^2) (q = 0.93kN/m^2) (q = 1.11kN/m^2)

Figure 4.12: Plastic Flow for All sides simply supported:



(q = 0.51 *kN/m*^{2}) (q = 0.69 *kN/m*^{2}) (q = 0.99 *kN/m*^{2})

Figure 4.13: Plastic Flow for All sides fixed support:



(q = 0.5 *kN/m*^{2}) (q = 1.5 *kN/m*^{2}) (q = 3.32 *kN/m*^{2})

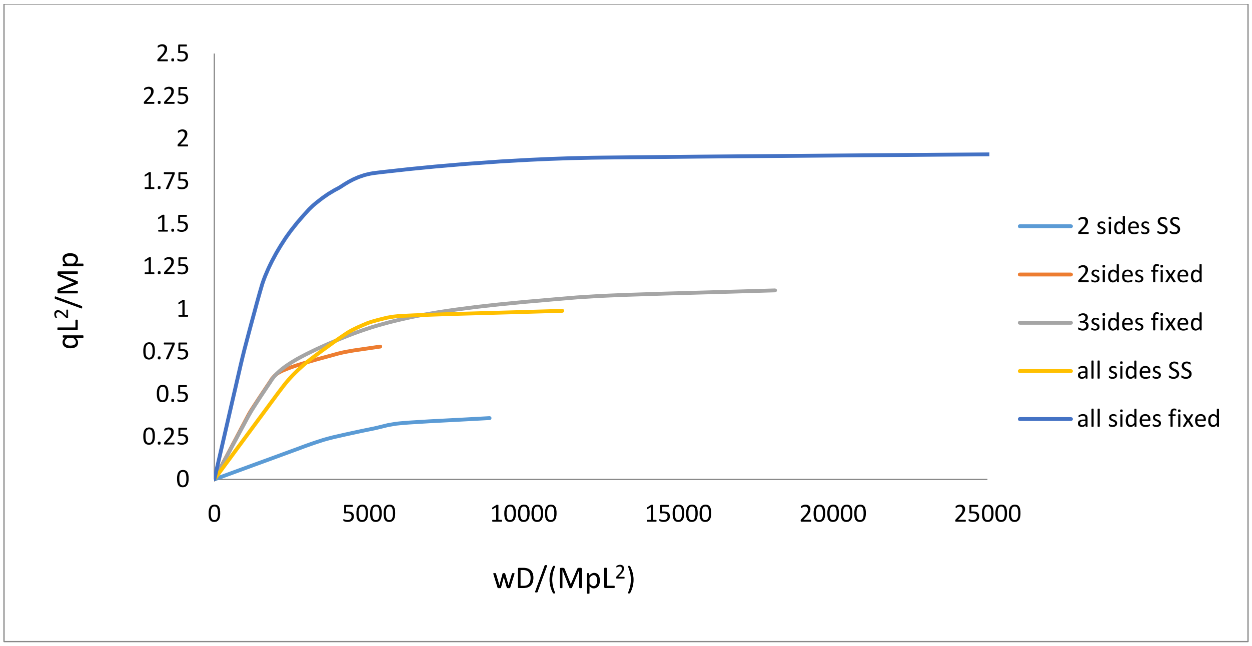


Fig 4.14: Comparison curve between load and deflection values for different boundary conditions

Boundary condition variation brings a conclusion that as ends are made free end, simply supported end and fixed end, the degree of freedom is restricted. So a higher value of load is required to cause the collapse. Hence plate with all sides fixed yields at much higher load value than the plate with two opposite sides simply supported and the other two sides free. So, increase in restrained degrees of freedom increase first yield load and collapse load.

4.3 Problem Discussion 2

The same problem is analyzed considering the strain hardening behavior of materials. Width and length of plate are kept as 1.0m each and thickness as 0.01m. A uniformly distributed 1.5kN/m^2 is applied over the plate surface. Other material properties are mentioned below.

Poison's ratio = ν =0. 3

Youngs modulus = E = 10.92 N/mm^2

$E_T = \frac{E}{2}$

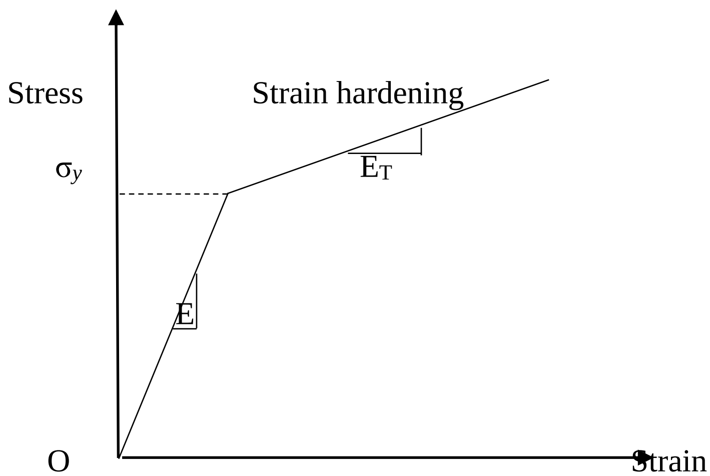
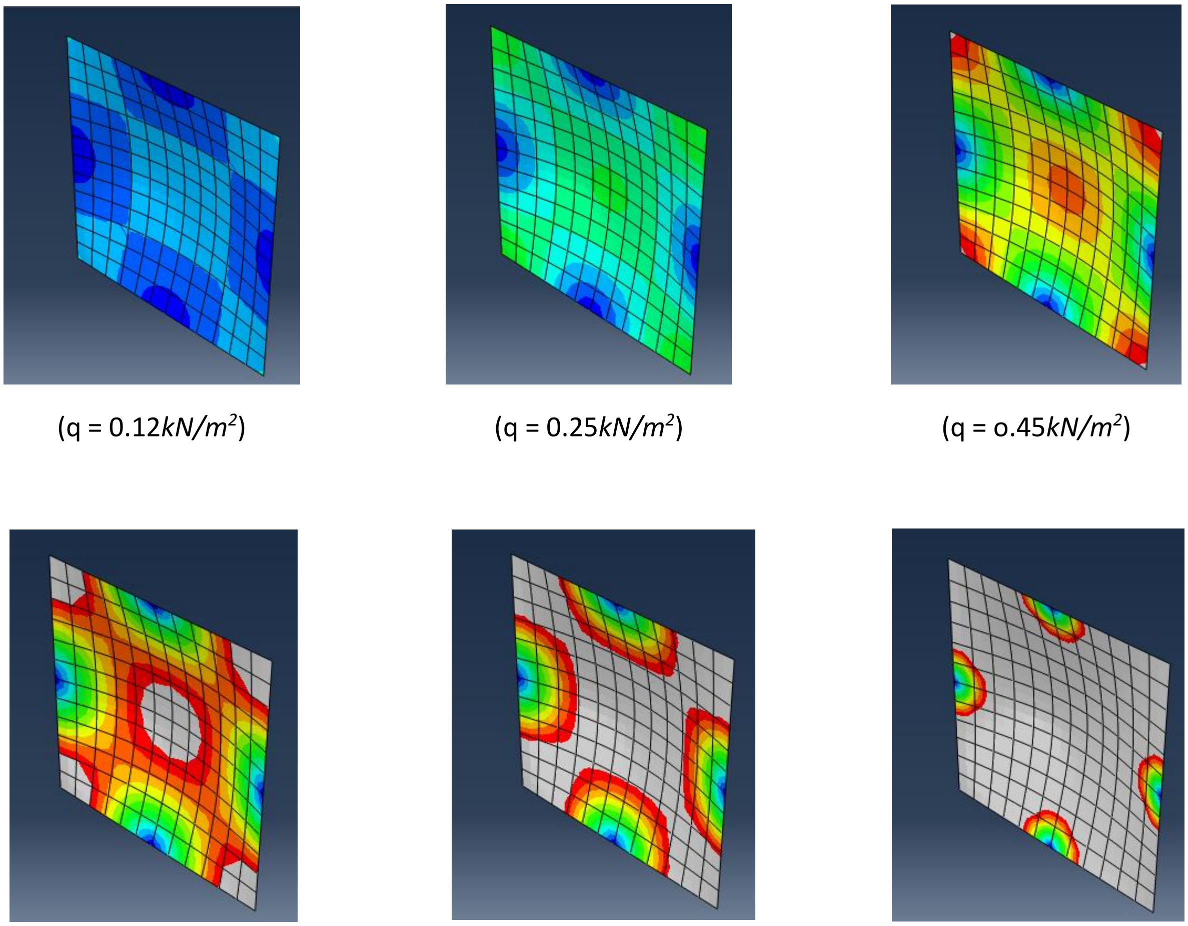


Fig 4.15: Constitutive Relation for Strain hardening material

The main objective of the work is to track the progressive plasticization of the cross sections. Since the stresses are calculated at different load steps, yielding of sections can be easily tracked which give an idea of plastic flow in the plate. The figures give an idea of plastic flow in simply supported plate. Due to strain hardening property, the plate will yield at a higher load. After yielding there will be a change in slope of stress–strain curve, whose value is taken as half of initial Youngs modulus value. Maximum deflection values at center of plate obtained from ABAQUS software and MATLAB coding are compared in the following table.

<u>Plastic flow pattern:</u>



(q = 0.12*kN/m*^{2}) (q = 0.25*kN/m*^{2}) (q = o.45*kN/m*^{2})

(q = 0.75*kN/m*^{2}) (q = 1.02*kN/m*^{2}) (q = 1.11*kN/m*^{2})

Fig 4.16: Stress flow pattern for plate having strain hardening property (ABAQUS)

The plastic flow followed the yield line pattern for s/s slab. The initial yielding started at corner points and further spread justifying the well-known corner lever effect in a simply supported two way slab used in yield analysis of slab. The white shows that because of strain hardening behavior of plate, the yield stress value exceeds the given yield value 1600*kN/m*^{2}.

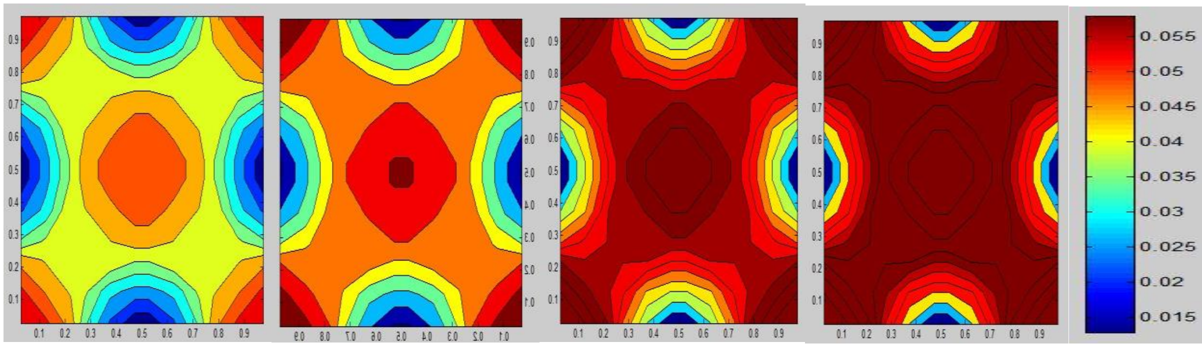


Fig 4.17: Stress flow in strain hardening material from MATLAB

A comparison curve between deflection values in plastic state and strain hardening state is plotted infig.4.18.

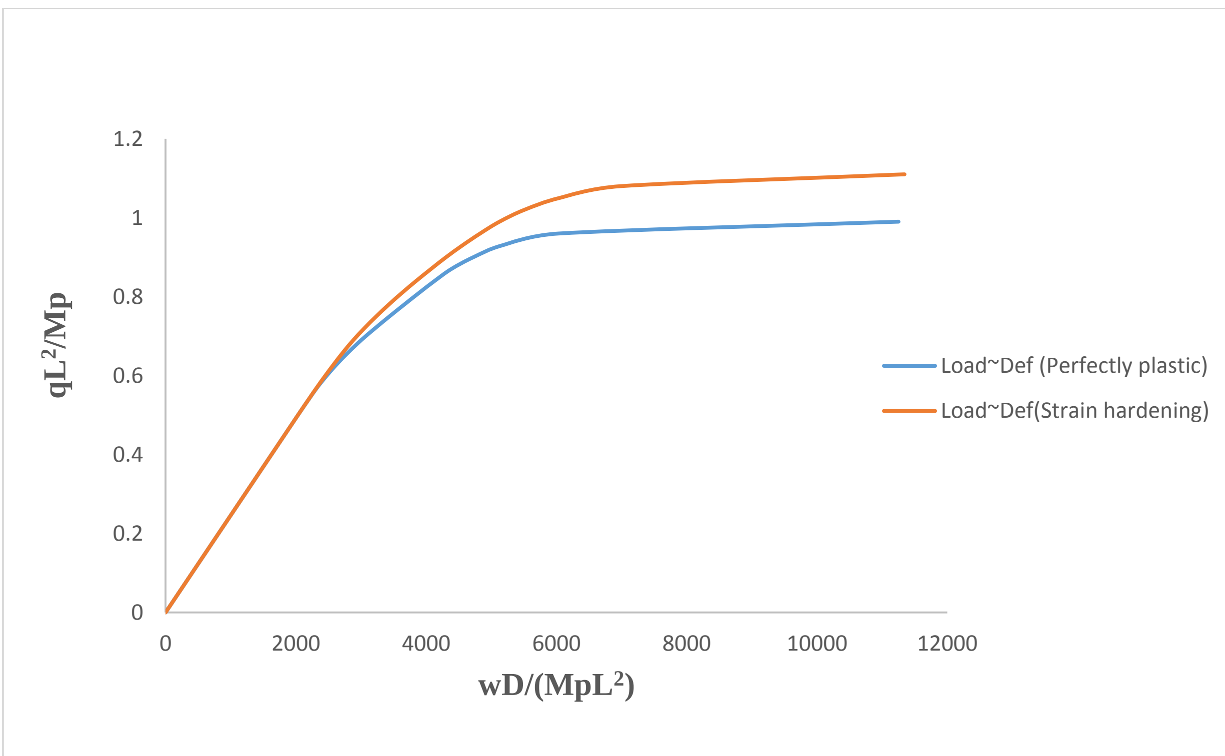


Fig 4.18: Load vs deflection curves for perfectly plastic and Strain hardening materials

From the figure it is clearly visible that up to elastic range both curves are identical, but beyond yielding due to strain hardening property, the plate takes a higher load value and finally collapses at 1.11kN/m^2 and almost follow the loading path.

CHAPTER 5

CONCLUSION

Chapter 5

Conclusion

The conclusions drawn from the present study are

1. The results obtained from ABAQUS are higher than results obtained from FEM based numerical method because it incorporate nonlinearity in flexure and shear both.

3. Lower yield loads and higher collapse loads are obtained from ABAQUS. Because a non-layered approach is used in formulations of FEM modeling. So first yield is observed when whole section has plasticized.

4. For plate thickness of 0.01m the influence of transverse shear on plastic behavior is small represented by difference in ABAQUS values and numerical values.

5. The influence of shear is observed to increase with increase in plate thickness.

6. The yield loads and collapse loads are observed to increase with increase in restrained degrees of freedom depending on the boundary conditions of plate.

7. Plastic flows are observed to follow the pattern as given by yield line theory for various boundary conditions.

8. The plastic flow pattern for strain hardening material clearly represents the isotropic strain hardening flow where the subsequent yield surfaces are a uniform expansion of the original yield curve. These are clearly visualized through the stress contour plots drawn in different stages of loadings in both ABAQUS and numerical method.

In problems dealing material non-linearity, convergence is required to satisfy the equilibrium condition and the stress conditions. Since analysis method follow step by step incremental approach with repetitive computations, this may lead to the error accumulations. Therefore the procedure is very sensitive and results depend on adopted incremental load (time step) and tolerance value.

References

1. Owen D.R.J. and Hinton E. (1980): Finite element in plasticity: Theory and Practice, Dept. of civil engineering, University College of Swansea, U.K., 121-152, 319-373.

2. Talja A. and Pekka S. (1992**):** bending strength of beams with nonlinear analysis, Rakenteiden Mekaniikka, 25, 50-67.

3. Shen Shen Hui (2000): Nonlinear bending of simply supported rectangular Reissner–Mindlin plates under the transverse and in-plane loads and resting on elastic foundations, Engineering Structure, 22, 847-856.

4. PawelWoelke (2005): computational model for elasto-plastic and damage analysis of plates and shells.

5. Szwaja Nicolas (2012)**:** Elastic and Elasto-Plastic Finite Element Analysis of a tension test specimen with and without voids, 15,147-175.

6. Fallah N. and Parayandeh A. (2014): A novel finite volume based formulation for the elasto-plastic analysis of plates, Thin-walled Structure, 77,153-164

7. ABAQUS/CAE manual